The software Saleae Logic 1.2.18

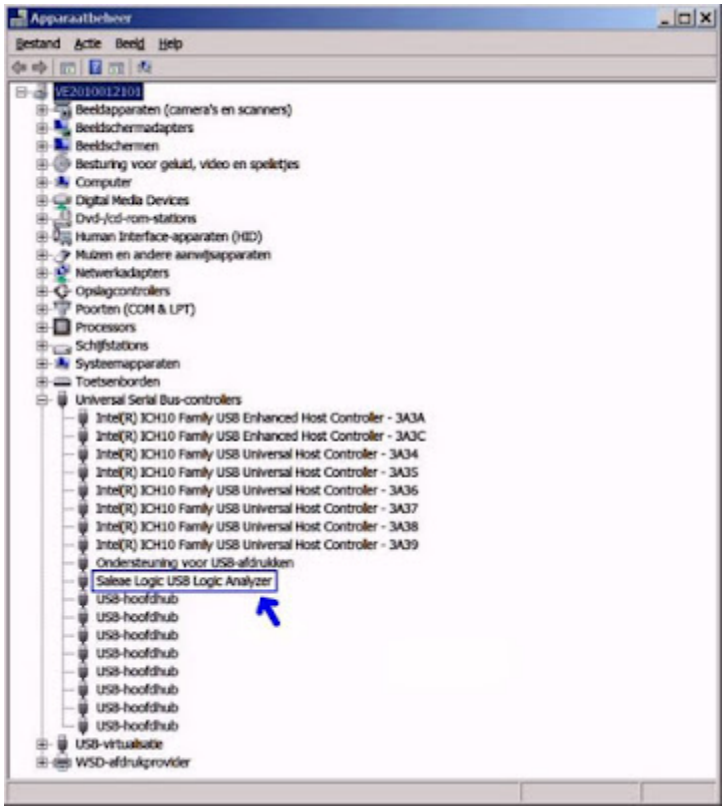
(Manual written by Jos Verstraten on 18/02/2019.)

Download and installation
The necessary software can be downloaded free of charge:

https://www.saleae.com/downloads/

The software is available for Windows, Linux and Mac OSx. You get a .EXE file of 118 MB called *'Logic Setup 1.2.18.exe'*.

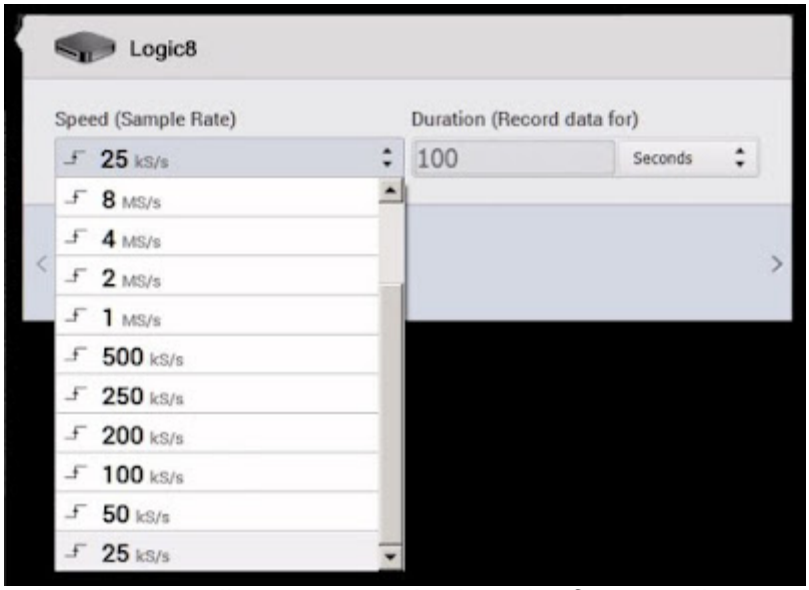
Installing the software goes entirely automatically. After the program *'Saleae Logic 1.2.18'* has been installed on your hard disk, the necessary USB driver is loaded. This driver will then automatically appear in the Windows Device Manager. If you connect the logic analyser to a USB port on your PC, Windows will automatically recognize the device.



The necessary USB driver is recognized by Windows.

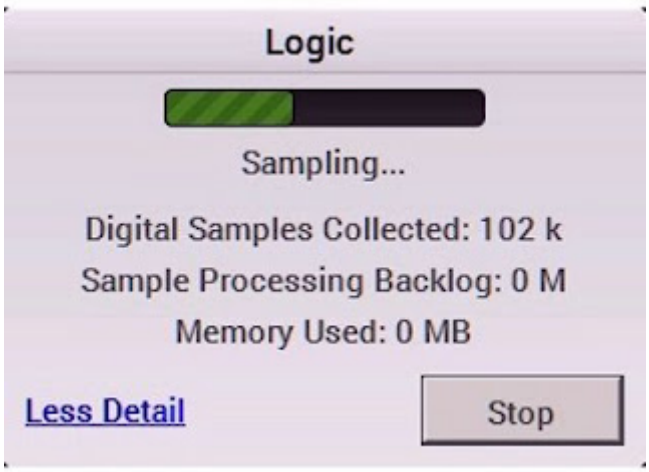
Connecting the logic analyser to your circuit
Connect the eight inputs of the logic analyser to eight logic signals. Don't forget to connect the ground and do so as the very first connection! As a test we connected the analyser to eight outputs of a CD4020 '14 stage ripple counter' which was clocked via a transistor stage from the signal of our function generator. The circuit was powered with 5 V_{dc} and the generator was set to 10 kHz square wave.
The manual warns for unforeseen ground loops that can occur and that result in a current through the ground connection of the analyser. This current can damage the device. It is recommended that you always make a second ground connection between the power supply of your circuit and the frame of your PC, for example via a second cable. Do not turn on the power of your circuit until you have connected the analyser to your PC via its USB cable and the device is therefore powered. You must connect this logic analyser directly to a USB 2.0 port of your PC and not via a USB hub or a docking station.

Working with the software
Open the software and connect the analyser to a USB 2.0 port on your PC using the supplied USB cable. After a few seconds the text *'Connected'* appears in the title bar of the application. At this moment, turn on the power of your circuit to be tested. Next to the *'Start'* button in the upper left corner of the window you will see a button with two up and down pointing arrows. If you click on it, the window of the figure below will appear. The software has recognized the analyser as a *'Logic8'* thanks to its Vendor ID and Product ID codes. In the left selection window you can set the speed or *'Sample Rate'* in 13 steps between 25 ksamples/s and 24 Msamples/s. In the right selection window you can set the size of the measuring cycle in samples, seconds or milliseconds.



Setting the sampling rate and the length of a sampling cycle.

Performing a sampling cycle
All you have to do is click the *'Start'* button. The logic analyser scans the eight inputs for each sample and determines whether the inputs are 'L' or 'H'. This data is collected in the memory of your PC in a temporary file. The window below appears on your screen, where you can follow the collection of the samples and, if you wish, interrupt the process.



Collecting the samples.

Display of all collected samples
After all the samples you have set have been collected or the sample time you have set has elapsed, that window disappears from your screen and all the collected samples are displayed at once. The scroll-bar at the bottom of the window allows you to scroll through the recorded samples. You can also do this by pressing the *'N'* and *'P'* keys on your keyboard. If you move your mouse to the sample window, you can use the scroll wheel of your mouse to increase or decrease the visible part of the samples. The same happens when you press the *'+'* or *'-'* keys on your

keyboard.

The display of the logic levels at the eight inputs.

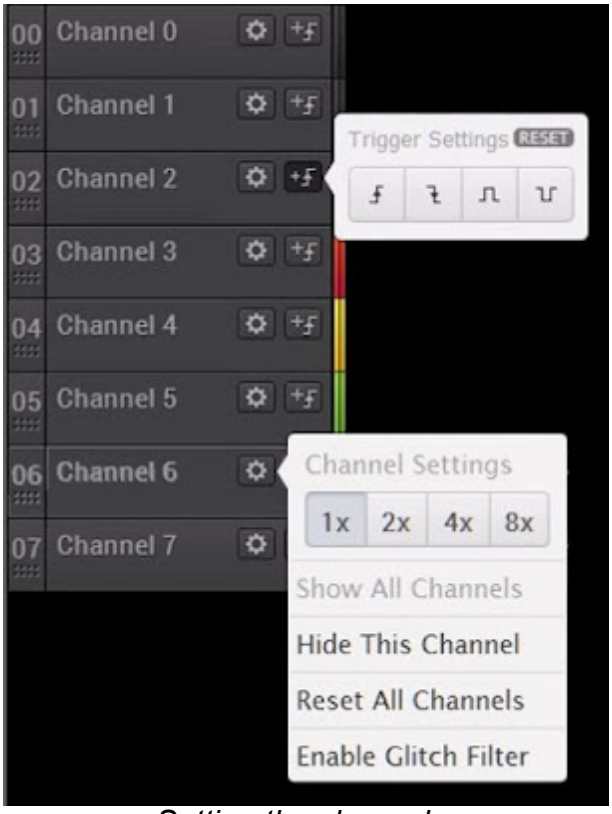
Save the collected samples
You can save the samples in a new tab by clicking on the '>>' box next to the *'Capture'* tab. You can change the name of the tab by hovering your mouse over the tab name and double-click. Afterwards you start with an empty tab and you can start a new measurement.

Saving the collected samples on individual tabs.

Setting the coloured traces
All eight traces are displayed in white. However, you can switch to a color display. Go to *'Options'* (top right) and open *'Preferences'*. Go to the *'Interaction'* tab and check the *'Use Color'* option.

Setting the channels

The menu for setting the channels
The eight channels are defined by a channel number and a channel name. The number obviously refers to the channel numbers on the input connector of the logic analyser. Remember that what in the software is *'Channel 0'* on the device can be *'Channel 1'*! The figure below shows which settings you have per channel.



Setting the channels.

Change order
If you move the mouse over the channel number area, the cursor changes into a hand and you can drag the channel to another location.

Changing the channel name
If you double-click with the mouse in the field with the channel name you can change it.

The Channel Settings
This setting allows you to increase the amplitude of the channel by a factor of 2, 4 or 8. With *'Hide This Channel'* the channel disappears from the screen. With *'Show All Channels'* all channels are displayed again. *'Reset All Channels'* restores the channel display to its original values, including order and names.

The trigger settings

Trigger conditions
If you do not set a trigger condition, the software will start registering the samples when you click *'Start'*. With the trigger settings, however, you can link the start moment of recording the samples to the signal flow on one or more channels. You can set per channel:

- Triggering on a leading edge.
- Triggering on a trailing edge.
- Triggering at a high level with an adjustable time.
- Triggering at a low level with an adjustable time.

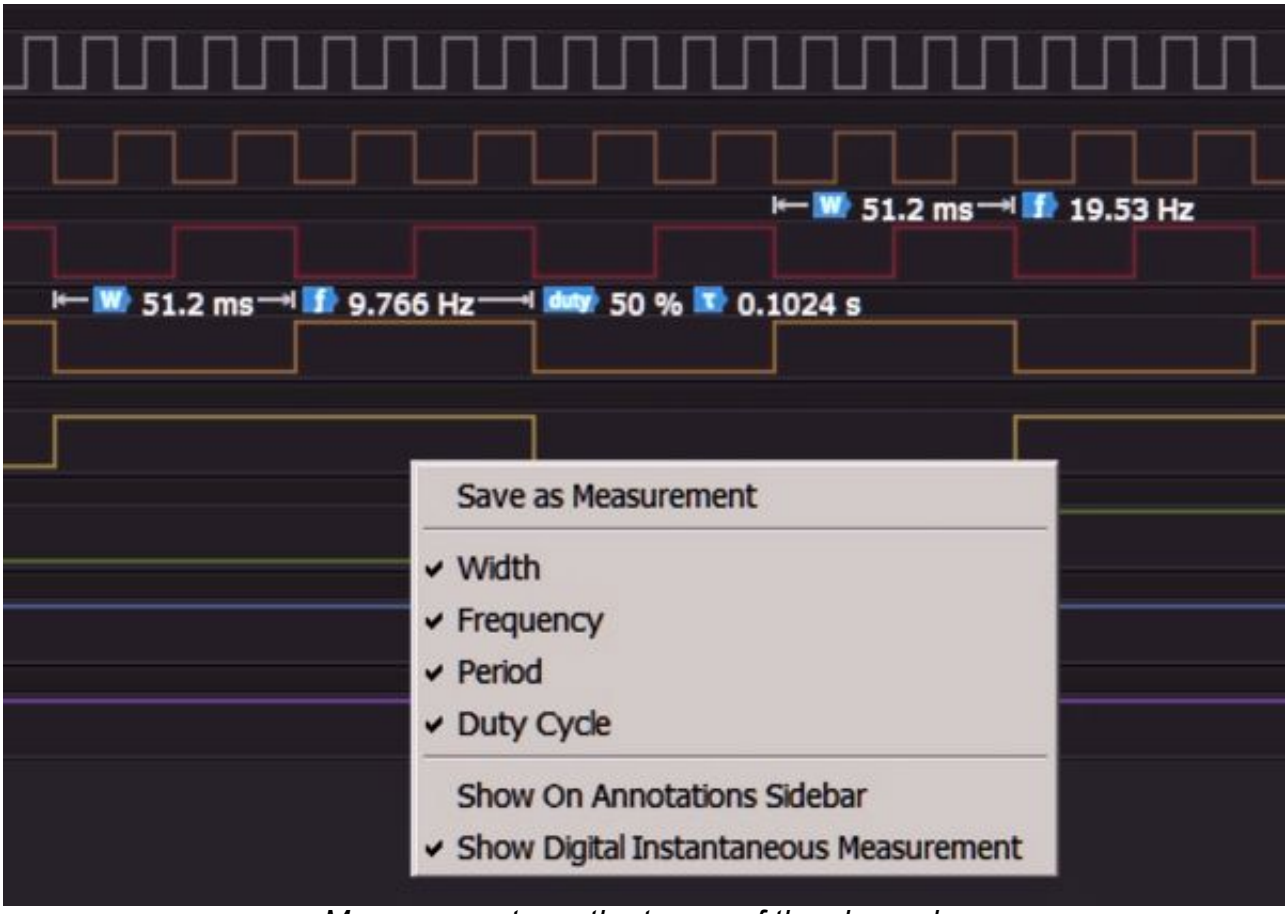
The last two options allow you to set a minimum and/or a maximum time in ns, µs, ms and s.
If you set the trigger conditions of one channel, the trigger conditions of all other channels are set to *'Don't Care'*, a cross will appear in the *'Trigger Settings'*. You can of course combine the trigger conditions of various channels. If you have set a trigger condition on one channel, you can only set an 'L' or 'H' as trigger condition on the other channels, without further specification.

Performing measurements on the traces

Immediate display of the signal parameters
Click with the right mouse button on one of the traces. In the popup window, select which parameters of the signal you want to measure:

- Width
- Frequency
- Period
- Duty Cycle

Click on *'Show Digital Instantaneous Measurement'*. If you now move to a trace you will immediately see the checked parameters appear for the selected period of the trace.



Measurements on the traces of the channels.

Annotations
The right column of the software window contains three words:

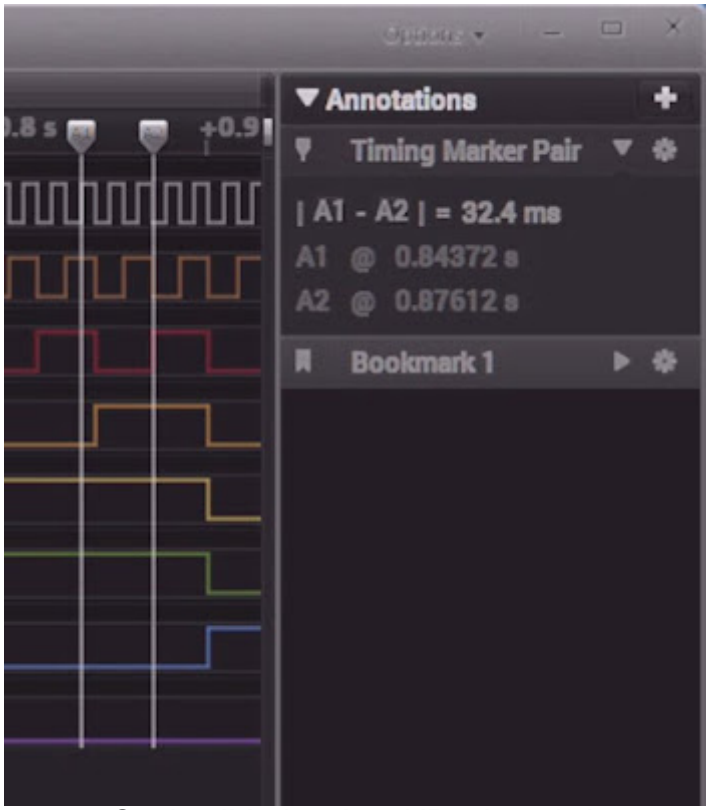
- Annotations.
- Analysers.
- Decoded protocols.

By clicking on *'Annotations'* you can display measurement data in this right column. Click on '+' and you will get three choices:

- Timing Marker pair.
- Bookmark.
- Measurement.

Timing Marker Pair
With this option you can mark two places in the data. After clicking on this option, this notation appears in the *'Annotations'*:
| A1 - A2 | = ####
Click on *'A1'* and move the cursor to where you want to place this marker. Repeat for *'A2'*. The markers are represented by two vertical lines in the traces and the *'Annotations'* column shows the time difference between the two markers. You can add multiple markers to the dataset. The same result is obtained by moving the cursor to the desired location and then pressing one of the number keys on your keyboard.

Bookmarks
With this option you can assign a name to a magnified part of the data. Afterwards, you can click on the name of the bookmark to view the magnified part of the dataset again.



Setting 'Markers' and 'Bookmarks'.

Measurement
When you click this option you will see a list of the parameters you can measure:

- Width.
- Frequency.
- Average Frequency.
- Average Duty Cycle.
- Count of Rising Edges.
- Count of Falling Edges.
- Count of Positive Pulses.
- Count of Negative Pulses.
- Minimum Positive Pulse Width.
- Maximum Positive Pulse Width.
- Minimum Negative Pulse Width.
- Maximum Negative Pulse Width.
- Period.
- Average Period.
- Count of Complete Periods.

Data saving and loading

Save or load a tab with data
'Options' and *'Save capture'* allow you to save the current tab with traces for future use. Via the same menu, but then the option *'Open capture/setup'* you can put an old set of data in a tab again.

Exporting the data
'Options' and *'Export data'* allows you to export your measured samples to various formats that can

be read in other programs, such as CSV, Binary, VCD and MatLab.

Protocol analysers

What are Protocol analysers?
'Protocol analysers' decode measured data that is encoded according to a certain protocol. The software currently offers no less than 24 protocol analysers:

- Async Serial
- I^2C
- SPI
- 1-Wire
- Atmel SWI
- BiSS C
- CAN
- DMX-512
- HD4478O
- HDLC
- HDMI CEC
- I^2S / PCM
- JTAG
- LIN
- MDIO
- Manchester
- Midi
- Modbus
- PS2 Keyboard/Mouse
- SMBus
- SWD
- Simple Parallel
- UNI/O
- USB LS

Fill in specific data
For each selected protocol, you must first fill in a window with specific data. However, this requires a thorough knowledge of the in's and out's of the chosen protocol, which goes beyond our rudimentary knowledge of these protocols.